Finite Elements in Analysis and Design 5 (1989) 111–118
Elsevier Science Publishers B.V., Amsterdam — Printed in the Netherlands

SOLUTION OF A LARGE STRUCTURAL ANALYSIS PROBLEM ON A MINICOMPUTER

S.K. BHARGAVA and Srikanth M. KANNAPAN *

Tata Research Development and Design Centre, 1 Mangaldas Road, Pune 1, India

M.M. GOSWAMI and R.J. SRINIVASAN

Tata Robins Fraser Ltd., 11 Station Road, Jamshedpur, Bihar, India

Received July 1987
Revised December 1988

Abstract. A large structural analysis problem, involving the solution of 11448 equations with an average front width of about 350, was solved on a minicomputer. This mathematical model resulted from the application of the finite element method, using flat shell elements, to determine the stress levels induced in the carriage of a stacker–reclaimer due to design loads. Computer programs were developed for analysis; the frontal solution procedure with a restart facility and other modifications were used for the solution of the finite element equations. A scheme was developed and employed to solve such large problems on a minicomputer in spite of the limited computing speed and disc storage space available. Seven load cases were analyzed and areas of high tensile and compressive stress were identified.

Introduction

Until recently, the utilization of finite element analysis was restricted to mainframe or super minicomputers. This had, to a great extent, restricted finite element applications to large organizations. Engineering work stations are increasingly becoming common in design offices. The engineering workstation is typically a 16–32 bit computer with a high resolution graphic display and other peripherals. Today there are a number of finite element packages that run on work stations and make use of its graphical capabilities to interactively preprocess and postprocess the finite element data. These packages have virtually opened the way to much more widespread employment of finite element techniques. However, minicomputers offer relatively slow computational speed and generally have limited disc storage capacities. Solutions of large structural analysis problems on such minicomputers require large computation time and judicious use of the available disc storage capacity for storing various back-up files required during analysis. These problems are further compounded by the fact that the complete system is available only for a limited period of time in a time sharing environment.

This paper describes the solution of a large structural analysis problem on a minicomputer. A scheme was developed and successfully employed to carry out the stress analysis of the carriage of a stacker–reclaimer machine, circumventing the practical difficulties of making available long uninterrupted slots of computing time and large amounts of disc space. This scheme enables a large structural analysis problem to be solved in a number of manageably short available time slots with considerably less disc storage requirements for most of the

---

* Currently at Department of Mechanical Engineering, University of Texas at Austin, Austin, Texas, U.S.A.

0168-874X/89/$3.50 © 1989, Elsevier Science Publishers B.V.

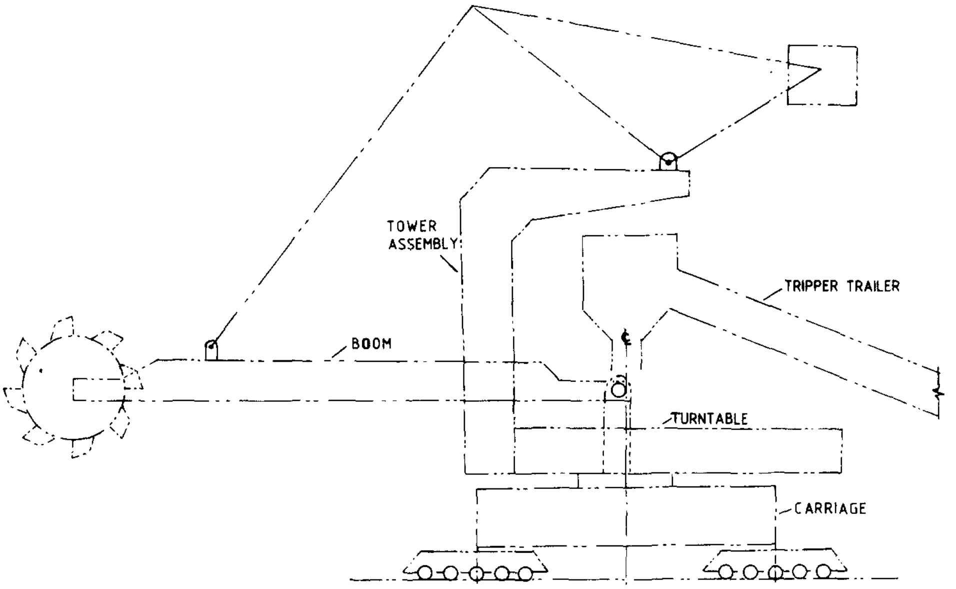

Fig. 1. General arrangement of stacker–reclaimer.

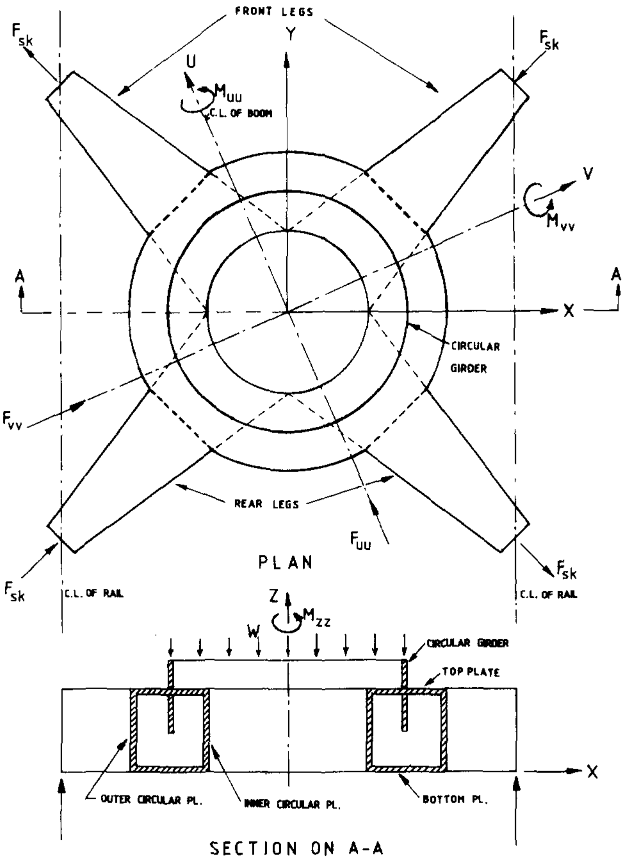

Fig. 2. Carriage assembly and prescribed loadings.

analysis time. The purpose of this work was to validate the design of the carriage of the stacker–reclaimer which has stresses induced in the structure under operating and nonoperating conditions. A stacker–reclaimer is a typical bulk material handling machine (Fig. 1) used in mines, power plants, ports, etc. The carriage of the stacker–reclaimer is mounted on a truck frame with wheels, moves over rails and supports the rotating and luffable parts of the machine. The carriage, fabricated from mild steel plates, is shaped like a torus (Fig. 2) with four legs and weighs about 13 tonnes. The structure was modeled using triangular flat plate elements. Seven load cases were analyzed and critical areas due to high tensile and compressive stresses were identified.

The analysis was carried out on an Omega 58000 minicomputer (similar to ICL Perq) running under the Unix operating system. The system configuration included 2 Megabyte (MB) RAM, 2 × 40 MB (32 MB formatted) hard disc. The available compiler did not support tape read/write from a Fortran program.

Finite element modeling

The carriage was modeled as a shell structure and idealized by an assembly of thin flat triangular plate elements. The plate elements are assumed to undergo both membrane and bending deformations.

Element characteristics

For the triangular element, at each node, the displacements u, v, w and rotations θ_x, θ_y, θ_z are taken as unknowns. Since it is assumed that the in plane forces do not affect the bending deformation and vice-versa, the equations governing element stiffness are obtained by adding individual stiffness matrices due to in plane and bending forces. While deriving stiffness matrices for in plane forces and bending forces, a linear variation is assumed for the in plane displacements and a cubic variation for the normal displacement. To avoid the difficulty of the singularity of the stiffness matrix when the elements are coplanar, as is the case in this analysis, a fictitious set of rotation stiffness coefficients are added in all the element stiffness matrices as suggested by Zienkiewicz [3].

Discretization

To simulate the complex and unsymmetric loadings on the structure, it became necessary to discretize the full structure even though the geometry is symmetric about the X and Y axes (Fig. 3). The structure is discretized into 4048 elements and defined by 1908 nodes. The discretization process required additional care due to intersecting plates in the model. Computer graphics was extensively used to check the nodal data and element connectivity. Figures 3 and 4 show the top plate and a view of the full model respectively. To show the internal details, a view of one quarter of the model is shown in Fig. 5.

Boundary Conditions

For each load case, operating loads specified were, superstructure weight, moments (M_{uu}, M_{vv}) along and perpendicular to the boom axis, slewing moment (M_{zz}) and forces in the X–Y plane consisting of shearing and skew forces. These are schematically shown in Fig. 2. Moments M_{uu} and M_{vv} were simulated by equivalent nodal forces acting in the Z direction, whereas M_{zz} was simulated by applying nodal loads on the legs in the X–Y plane. The super structure weight was distributed as nodal forces on nodes on top end of the girder. The other forces in the X–Y plane were equally distributed on nodes on the back face of the legs. The directions of some of these loads are dependent on the position of the boom with respect to the

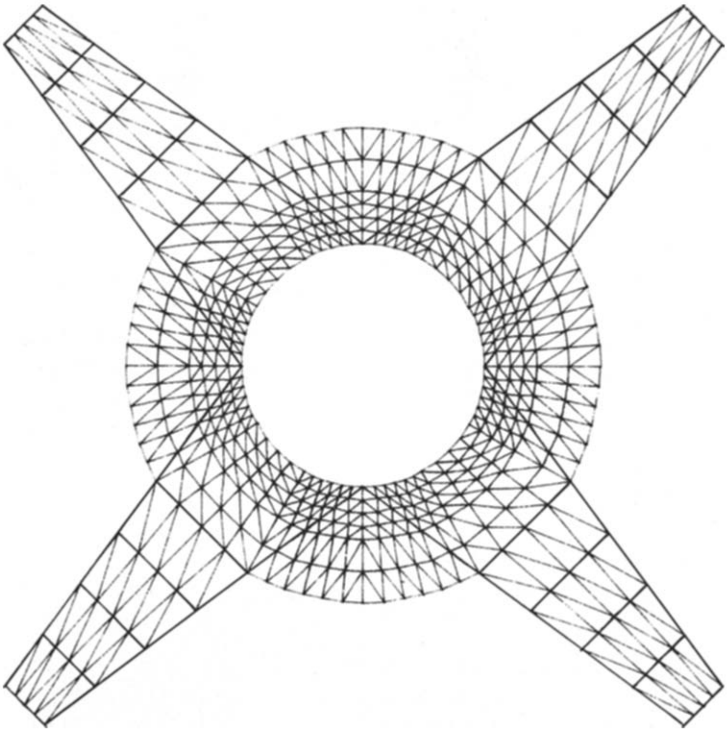

Fig. 3. Plan of carriage assembly discretization.

legs. Consequently, seven load cases are considered with different boom positions: four under normal operating conditions and three under non-operating windy conditions. The nodes on the top end of the circular girder are fixed in the X and Y directions and the nodes at the back of the front legs are fixed in the Z direction.

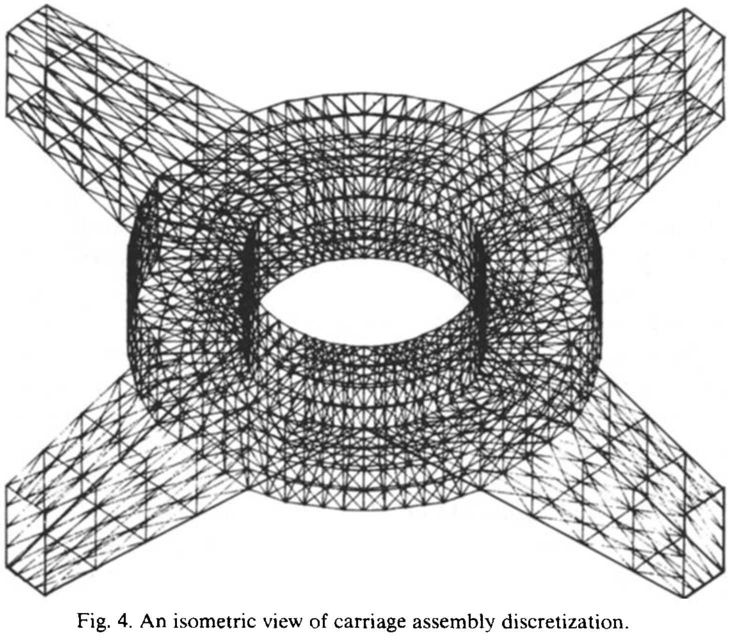

Fig. 4. An isometric view of carriage assembly discretization.

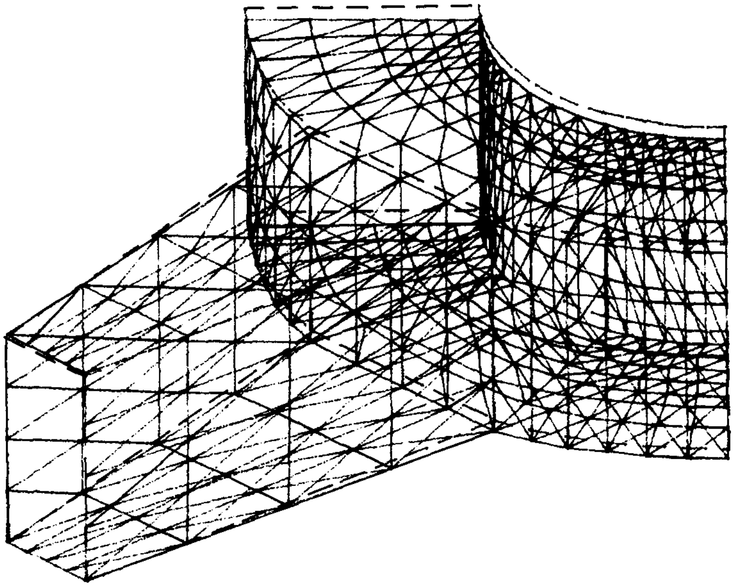

Fig. 5. Internal details of model with superimposed distorted shape.

In practice, the vertical reactions of the rear legs are kept equal by a hydraulic connection. To simulate this condition, equal forces are applied on the back of the rear legs. The magnitude of these equal forces are kept the same as the reactions on the rear legs in the Z direction. The vertical reactions are evaluated manually for each load case by considering the overall equilibrium of the carriage under the operating loads while assuming equal vertical rear leg reactions.

Solution procedure

The modeling approach described above resulted in a mathematical model of 11 448 linear equations. Since an in-core solution of such a large system of equations is not practicable even on main-frame computers, an out-of-core solution technique based on the frontal solution procedure [1] is used. The frontal solution procedure is selected because of its advantages over other schemes (e.g. banded and profile storage schemes) in terms of efficient use of high speed back-up storage, economy in solution time and ease of data preparation. These advantages are particularly true for closed structures like the carriage. The maximum front width encountered in the model is about 450 with an average front width of about 350. The frontal solution program presented in [1] was modified to take advantage of matrix symmetry. Other features, described in the following sections, were also included in the program to circumvent the problems related to slow computing speed and limited disc space on a minicomputer.

To avoid the requirement of a large uninterrupted slot of computing time, the analysis procedure was broken up into the steps given in Table 1. The steps are run separately in a given sequence so that each step is run in a manageable time slot with the output of the step being stored on hard disc. The stored output is used later for the next step. For the second step involving reduction of the global equations, the most time-consuming step in large problems, a restart facility is provided. The restart facility allows this step to be completed after running this step over a number of available time slots. After the analysis of the first load case the reduced matrix is saved on magnetic tape. Thus, resolution for other load cases can be carried

Table 1
Back-up space requirements at different steps

Step	Expected run time	I/O files	Total space required
1. Evaluation of stiffness and body force matrices.	2 hours	Fort.9 Fort.10	3.12 MB
2. Assembly and elimination of global equations.	50 hours	Fort 9. Fort.10 Fort.2 Fort.3 Fort.13 Fort. 13.bak	40 MB
3. Back substitution/resolution and evaluation of displacments (for each load case).	1.5 hours each	Fort.2 Fort.3	33 MB
4. Stress evaluation (for each load case).	2 hours each	Displa	1 MB

out separately after loading the saved file from tape to hard disc. The procedure described above was implemented by modifying the existing finite element package to write element stiffness matrices and body force (structure self weight) matrices in prescribed files. Under different user options the program can also evaluate stresses after reading nodal displacements from a given file. The restart facility in the frontal solution program was implemented by dumping the current assembled stiffness matrix and a few other parameters into a file every time after a user specified number of equations were eliminated. In the restart mode, the dumped data is reloaded and the program continues from the last stored position. The "prefront" data which encodes information about the order in which the nodes are to be eliminated, is calculated only once in the initial run and stored in a file. For the subsequent restart runs the 'prefront' data is read from the file to save time.

Considering the size of the problem, it became necessary to precalculate the storage requirement of the various input and output files at each step of the analysis to determine if sufficient disc space was indeed available. Table 2 gives the list of various major files which were required for the analysis. The estimate of maximum disc space requirement at each step is given in Table 1. A program was developed to predict the back-up storage requirement for the frontal solution procedure, given the element connectivity data of a finite element model. At the time of the analysis, the disc space available to users was distributed on two discs of 26 MB and 32 MB respectively. This meant that the size of any file residing on disc could not be more than 32 MB. Due to the large size of the reduced matrix, the back-up data for the frontal solution required 32 MB of disc space. Consequently, the back-up data had to be stored in two files, one on each disc. The reduced equations of the first one half of the equations were stored in the first file (Fort.2) and the rest of the equations in the second file (Fort.3). The frontal solution program was modified to store the reduced equations in two files in a user defined ratio and also to carry out back substitution accordingly.

Storing the reduced equations in two files offered the advantages of running the reduction step with considerably less disc space requirements and thus allowed this step, which is the most time consuming step in the analysis, to be run in a time sharing environment. This is possible since the equation reduction is done sequentially, after the prescribed number of equations are reduced and written in the first file it is no longer needed in the reduction step. The file can then be saved on tape and the freed space used for the second file. In this way the requirement for the maximum disc space was brought down from 40 MB to 23 MB in this step.

Table 2
Description of major files generated during analysis

File name	Description	Size
Fort.2	Reduced matrix of first half number of equations	16.70 MB
Fort.3	Reduced matrix of second half number of equations	16.27 MB
Fort.9	Element stiffness matrices	2.8 MB
Fort.10	Element self weight matrices	0.32 MB
Fort.13	Data required for restart facility	1.07 MB
Fort.13.bak	Back-up of Fort.13	1.07 MB
Noddat	Prefront data	0.64 MB
Displa	Displacement (per load case)	0.15 MB

By making use of the restart capability after loading the frontal back-up file on the disc, the back substitution step is carried out for a number of load cases. The step is carried out in single user mode as it requires only about 34 MB of disc space and 1.5 hours of CPU time per load case. Thus, this scheme not only allowed one to overcome the disc space limitations but also enabled the analysis to be run in time sharing environment for the most part without unduly affecting other users.

Results and discussions

The analysis results predicted the behavior of the carriage consistent with the loading pattern. The rounding-off error, measured based on the X, Y and Z sum of reactions and the sum of applied X, Y and Z forces comparison, was found to be less than 1.1% ($X \sim 0.4\%$, $Y \sim 0.8\%$ and $Z \sim 1.1\%$). Similarly, a comparison was made between responses of pairs of symmetrically located nodes about plane of real symmetry, under symmetric boundary conditions to check error associated with the choice of element model and the number of degrees of freedom. This was found to be within 0.8%.

Seven load cases based on different positions of the boom were analyzed under operating loads and wind loads. In addition to the nodal displacements, principal stresses were evaluated on both sides of the element at the centroid. For each load case a number of plots were obtained to display the analysis results by superimposing the deformed shape (exaggerated) over the original shape of the structure using a specially developed post-processor (Fig. 5).

The analysis results show predominantly membrane action in the structure. The analysis predicts that generally, the top and inner circular plate are under compression. Critical areas for high tensile stresses are found to be (1) in the inner circular plate near the bottom and (2) in the bottom plate around the junctions of the cross plates and the outer circular plate on the side of the front leg. The circular girder is found to be most critical from the point of view of compressive stresses. Though the stresses in the structure were found to be within the permissible limits, the analysis also identified a few areas where stresses were found to be consistently low for the load cases considered. In these areas, material saving may be considered by reduction of plate thickness.

Conclusion

A scheme was developed and successfully employed to overcome the computational resource limitations in terms of computing speed and disc space to enable the solution of a large stress analysis problem on a minicomputer. An available finite element package was modified and a frontal solution procedure with a restart facility was developed. Computer programs implementing the scheme allowed the analysis which would have required large uninterrupted computing time to be run instead over a number of available time slots with considerably reduced disc space requirements for most of the analysis time. Using these programs, the carriage of a stacker-reclaimer machine was modeled and the resulting 11 448 linear equations with a bandwidth of approximately 350, was solved.

The stress analysis for seven load cases was carried out and the areas of high and low stresses were identified. The analysis results were used in validating the design of the carriage. The computer programs developed are general purpose and can be used in the design-analysis of other components of the stacker–reclaimer (e.g. boom, turntable) and also other structures.

Acknowledgments

The authors thank Dr. E.C. Subbarao, Director, Tata Research Development & Design Centre and Mr. F.S. Tarapore, Chairman, Tata Robins Fraser Ltd. for their encouragement and suggestions throughout the execution of this work. The authors also thank the management of Tata Robins Fraser Ltd. for funding and permitting the authors to publish this work.

References

[1] Hood, P., "Frontal solution program for unsymmetric matrices", *Int. J. Numer. Methods Eng.* **10**, 1976.

[2] Melosh, R.J. and R.M. Bamford, "Efficient solution of load deflection equation", *J. Struct. Div. ASCE* **ST4**, pp. 661–678, April 1969.

[3] Zienkiewicz, O.C., *The Finite Element Method*, Tata McGraw Hill, New Delhi, 1979.